

2-16-2017

Incorporation of composite defects from ultrasonic NDE into CAD and FE models

Onur Rauf Bingol

Iowa State University, orbingol@iastate.edu

Bryan Schiefelbein

Iowa State University

Robert J. Grandin

Iowa State University, rgrandin@iastate.edu

Stephen D. Holland

Iowa State University, sdh4@iastate.edu

Adarsh Krishnamurthy

Iowa State University, adarsh@iastate.edu

Follow this and additional works at: https://lib.dr.iastate.edu/cnde_conf

 Part of the [Materials Science and Engineering Commons](#), and the [Structures and Materials Commons](#)

The complete bibliographic information for this item can be found at https://lib.dr.iastate.edu/cnde_conf/123. For information on how to cite this item, please visit <http://lib.dr.iastate.edu/howtocite.html>.

Incorporation of composite defects from ultrasonic NDE into CAD and FE models

Abstract

Fiber-reinforced composites are widely used in aerospace industry due to their combined properties of high strength and low weight. However, owing to their complex structure, it is difficult to assess the impact of manufacturing defects and service damage on their residual life. While, ultrasonic testing (UT) is the preferred NDE method to identify the presence of defects in composites, there are no reasonable ways to model the damage and evaluate the structural integrity of composites. We have developed an automated framework to incorporate flaws and known composite damage automatically into a finite element analysis (FEA) model of composites, ultimately aiding in accessing the residual life of composites and make informed decisions regarding repairs. The framework can be used to generate a layer-by-layer 3D structural CAD model of the composite laminates replicating their manufacturing process. Outlines of structural defects, such as delaminations, are automatically detected from UT of the laminate and are incorporated into the CAD model between the appropriate layers. In addition, the framework allows for direct structural analysis of the resulting 3D CAD models with defects by automatically applying the appropriate boundary conditions. In this paper, we show a working proof-of-concept for the composite model builder with capabilities of incorporating delaminations between laminate layers and automatically preparing the CAD model for structural analysis using a FEA software.

Keywords

Ultrasonic testing, Composite materials, Representation theory, Industry, Finite-element analysis, Composite models

Disciplines

Materials Science and Engineering | Structures and Materials

Comments

This proceeding may be downloaded for personal use only. Any other use requires prior permission of the author and AIP Publishing. This proceeding appeared in Bingol, Onur Rauf, Bryan Schiefelbein, Robert J. Grandin, Stephen D. Holland, and Adarsh Krishnamurthy. "Incorporation of composite defects from ultrasonic NDE into CAD and FE models." *AIP Conference Proceedings* 1806, no. 1, (2017): 150004. , and may be found at DOI: [10.1063/1.4974728](https://doi.org/10.1063/1.4974728). Posted with permission.

Incorporation of composite defects from ultrasonic NDE into CAD and FE models

Cite as: AIP Conference Proceedings **1806**, 150004 (2017); <https://doi.org/10.1063/1.4974728>
Published Online: 16 February 2017

Onur Rauf Bingol, Bryan Schiefelbein, Robert J. Grandin, Stephen D. Holland, and Adarsh Krishnamurthy



View Online



Export Citation

ARTICLES YOU MAY BE INTERESTED IN

UTSim2 validation

AIP Conference Proceedings **1806**, 150007 (2017); <https://doi.org/10.1063/1.4974731>

Numerical simulation of ultrasonic wave propagation in fiber reinforced plastic using image-based modeling

AIP Conference Proceedings **1806**, 150005 (2017); <https://doi.org/10.1063/1.4974729>

Surrogate modeling of ultrasonic simulations using data-driven methods

AIP Conference Proceedings **1806**, 150002 (2017); <https://doi.org/10.1063/1.4974726>

Lock-in Amplifiers up to 600 MHz

starting at

\$6,210



 Zurich Instruments

Watch the Video 



Incorporation of Composite Defects from Ultrasonic NDE into CAD and FE Models

Onur Rauf Bingol^{1,3,b)}, Bryan Schiefelbein^{2,3,c)}, Robert J. Grandin^{3,d)}, Stephen D. Holland^{2,3,e)}, and Adarsh Krishnamurthy^{1,3,a)}

¹*Department of Mechanical Engineering, Iowa State University, Ames, IA, 50011, USA*

²*Department of Aerospace Engineering, Iowa State University, Ames, IA, 50011, USA*

³*Center for Non-Destructive Evaluation, Iowa State University, Ames, IA, 50011, USA*

^{a)}Corresponding author: adarsh@iastate.edu

^{b)}orbingol@iastate.edu

^{c)}bryans70@iastate.edu

^{d)}rgrandin@iastate.edu

^{e)}sdh4@iastate.edu

Abstract. Fiber-reinforced composites are widely used in aerospace industry due to their combined properties of high strength and low weight. However, owing to their complex structure, it is difficult to assess the impact of manufacturing defects and service damage on their residual life. While, ultrasonic testing (UT) is the preferred NDE method to identify the presence of defects in composites, there are no reasonable ways to model the damage and evaluate the structural integrity of composites. We have developed an automated framework to incorporate flaws and known composite damage automatically into a finite element analysis (FEA) model of composites, ultimately aiding in accessing the residual life of composites and make informed decisions regarding repairs. The framework can be used to generate a layer-by-layer 3D structural CAD model of the composite laminates replicating their manufacturing process. Outlines of structural defects, such as delaminations, are automatically detected from UT of the laminate and are incorporated into the CAD model between the appropriate layers. In addition, the framework allows for direct structural analysis of the resulting 3D CAD models with defects by automatically applying the appropriate boundary conditions. In this paper, we show a working proof-of-concept for the composite model builder with capabilities of incorporating delaminations between laminate layers and automatically preparing the CAD model for structural analysis using a FEA software.

INTRODUCTION

Fiber-reinforced composite materials, especially carbon fiber reinforced composites (CFRP), are being used in automobile and aircraft industries due to their comparably low weight and impressive mechanical properties. Recent developments in composite production allow replacement of the structural elements of high performance air and ground vehicles with composite counterparts. A good example of these developments is the fully composite wings and fuselage of the Boeing 787 Dreamliner. Due to their increasing use in mission-critical parts of such vehicles, it is important to assess the residual life of composites due to production defects and in-service damage. Ultrasonic Non-Destructive Evaluation (NDE) is the preferred method for identifying many defects such as delaminations in composites. Although ultrasonic testing (UT) can be used to identify the presence of such defects, to determine the residual life, the damage need to be modeled and the structural integrity of the composite in the presence of defects need to be evaluated. Defect incorporation into 3D structural models [1] and several NDE data fusion methods for damage inspection [2] have been studied previously. However, currently, there are no reasonable methods to model the damage and evaluate the structural integrity of the composites.

In this paper, we describe an automated framework to incorporate defects measured using ultrasonic testing (UT) into finite element (FE) models of the composites and perform structural analysis, as illustrated in Figure 1. We create a concrete 3D CAD representation of the defect within a composite laminate structure using state-of-the-art CAD solid modeling kernels. The multiple laminae in the structure are generated using configurable layers, represented using

non-uniform rational basis spline (NURBS) surfaces, and include additional information such as the layup directions. The data from UT scans is segmented to identify the shape of the delaminated region, and is incorporated into the CAD model by splitting the corresponding surface of the lamina into separate faces. The resulting CAD model is prepared for FEA by applying contact boundary conditions to the delaminated faces and cohesive boundary conditions to the intact lam modeling software :

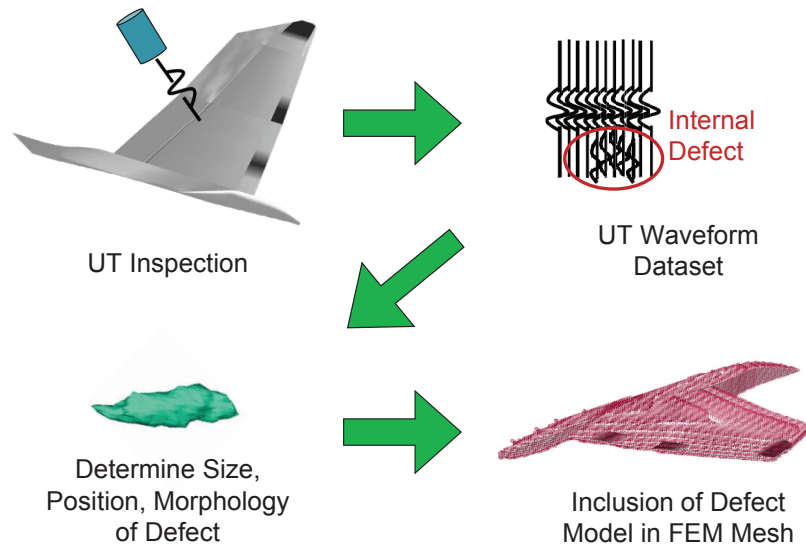


FIGURE 1. Illustration of the steps required to incorporate defects into the finite element tools.

METHODS

The automated framework consists of a model generator and analysis library, which can be easily customized using a high-level scripting language. The framework can be used to process NDE data and incorporate them into 3D models of the composite laminate. The framework also includes methods to automatically prepare the model for structural analysis using FEA by applying the appropriate cohesive and contact boundary conditions between the layers of the laminate.

Components of the Framework

The automated framework consists of two main components, the CAD Model Builder and FE Model Builder.

1. **CAD Model Builder:** The CAD Model Builder is designed to create 3D structural representations of the composite laminate and its layered structure, incorporate defects between the chosen layers and calculate required boundary condition information for the FEA software. The layers can be created using a NURBS surface, which is an industry-standard for representing geometrical curves and surfaces in CAD systems, or from the top or bottom surfaces of the existing layers.
2. **FE Model Builder:** The FE model builder can use the composite laminate models generated by the CAD model builder to perform structural analysis. The FE model builder directly imports the CAD geometry (including the defects), sets the appropriate boundary conditions between the different layers, and applies external forces.

Defect Modeling using the Cohesive Elements Model

One of the main defects that can be detected in the composite laminates using UT is delamination, which is the separation of the laminae/ /due to repeated stresses caused by composites' operating conditions [5]. The initial focus of our automated framework is to create and analyze delaminated 3D structural models of the composites. The framework uses Cohesive Elements Model [6] to test the delaminated structures by focusing on analyzing the bonding between different laminae. An example of the use of Cohesive Element Model in delaminations is shown in Figure 2. The bonded regions are modeled using a force-displacement relation between the laminae of the composite. A contact boundary condition is applied to the delaminated regions which prevents the interpenetration of the surfaces of the lamina in the delaminated regions while allowing for the lamina to separate freely. To achieve better convergence of the 3D structural models, a region of free boundary without any cohesive or contact model is necessary. This region is also meshed using appropriately small elements to allow for delamination growth in dynamic simulations.

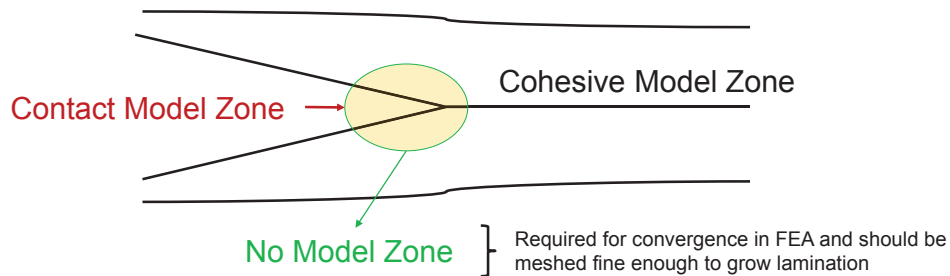


FIGURE 2. Illustration of the Cohesive Elements Model used for delamination analysis.

Processing NDE Data to Identify Delaminations

Accurate estimation of the residual strength of composites requires the incorporation of the defect data into the 3D structural models of composite laminates. The NDE data obtained from the UT scans provide an indication of defects, such as delaminations, which need to be mapped onto their correct spatial locations within the structural model. This is achieved by processing the raw UT scan data with a series of processing steps, illustrated in Figure 3.

First, the signal envelope is calculated so that all further analysis is based on reflection magnitude and errors due to waveform-phase are minimized. Next, the sample's front surface echo is aligned, making the front surface planar. After the front-surface echo is aligned, internal delaminations are detected by locating amplitude peaks that occur after the front surface. This alignment process is shown in Figure 3(a).

The time at which these subsequent peaks occur is directly related to the delamination depth and this time-of-flight information is used to assign a ply identification number to each peak. Currently, this analysis only considers the first peak that occurs after the front-surface reflection. After a ply ID has been established for each scan point, a 2D binary array, which identifies the presence of a delamination, can be created for each ply. This is shown in Figure 3(b). The top image in Figure 3(b) shows the scanned region colored by ply ID. The lower image shows a binary data set of a single ply.

In their raw form, the binary data sets are not suitable for incorporation into the finite element solver. The binary data is often noisy, as shown in the lower portion of Figure 3(b), and contains a level of detail which causes the finite element solution to become time-prohibitive. The binary data is cleaned through a pair of erosion/dilation operations. First the data is eroded, followed by dilation, so that small features are removed. Then, the data is dilated, followed by erosion, to fill internal holes. The filter kernel sizes associated with these operations are chosen based on the desired threshold for minimum feature size. Finally, a contour is extracted from the cleaned binary data and then imported into the solid modeler so that proper inter-ply boundary conditions can be set. These final processing steps are shown in Figure 3(c). These processing steps are implemented using a collection of custom plugins developed for the open-source data visualization tool ParaView [7].

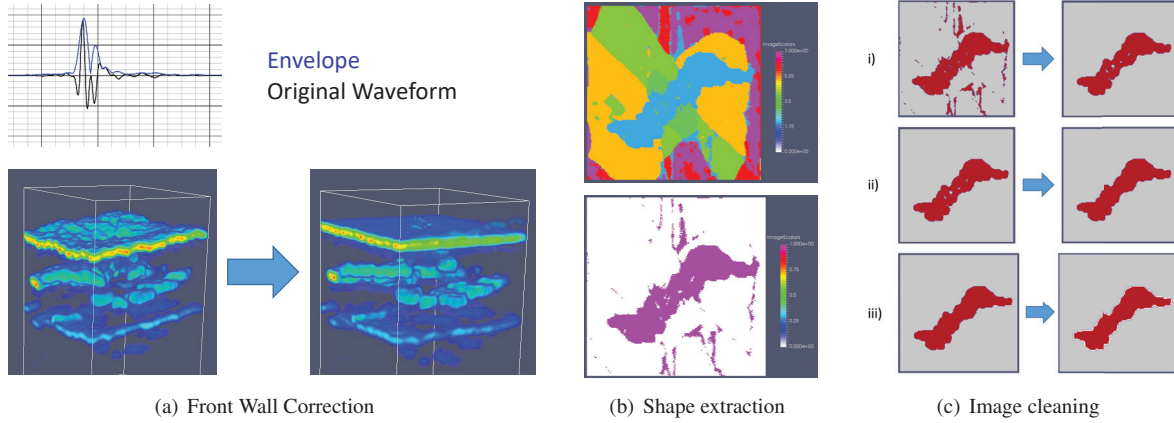


FIGURE 3. Illustration of the steps for delamination shape extraction. Processing begins by calculating the waveform envelope and aligning the front surface echo in time (a). Inter-ply delaminations are then identified by the subsequent echos, and each delamination is assigned a ply identification number to associate it with a particular ply. The scan region can then be visualized as a function of the ID number, and a binary data set may be generated for each ply (b). Finally, the binary data is cleaned using erosion/dilation operations in order to obtain a boundary-defining contour suitable for use with the finite-element solver (c).

Software Framework

The automated framework is developed using C++ [8] and Python [9] programming languages. The automated framework has a Python interface for user interactions. The two major components of the framework are implemented as follows:

1. **CAD Model Builder:** The CAD Model Builder uses a commercial solid modeling kernel, ACIS, to generate the laminae and incorporate the defects. ACIS is a geometric modeling library with C++ functions. This library is wrapped with an easy-to-understand lightweight C++ API designed for use with layered composite structures. This allows users to operate the CAD Model Builder without any knowledge of the solid modeling kernel internals. This lightweight API is converted into a Python module using an open-source tool, SWIG, to wrap C++ functions to use in Python [10]. The CAD Model Builder can also be extended to use different commercial or open-source solid modeling kernels.
2. **FE Model Builder:** The Finite Element (FE) Model Builder is specifically designed to work with SIMULIA Abaqus FEA software suite using its Python programming interface. The FE Model Builder uses the solid CAD model generated by the CAD model builder and represents it internally as an Abaqus object. The programming interface is used to apply the appropriate boundary conditions, external loading, material properties, mesh the model, and run the analysis.

The defect information used in the CAD Model Builder can be obtained from the real composite samples using UT scans or simulated using a software, such as UTSim [11].

The steps of the delamination analysis defined in the automated framework is illustrated in Figure 4. We construct a CAD representation of the intact composite model and then incorporate the delaminations between the surfaces of the composite CAD model. The framework replicates the laminate manufacturing process to create the CAD model of the composite. A NURBS surface is used to define the mold on which the composite layers are laid. Using this mold surface, along with a thickness value, a single layer of the composite can be created. Our framework can then extend on this layer by using the top or the bottom surfaces of the previously-created layers as a new mold. Using this method a composite consisting of multiple layers can be created.

After creating the CAD model of the composite laminate, the CAD model builder can add defects to the layers. Currently, since our initial focus is on delaminated composites, the framework can segment the faces of the surfaces between the different layers based on the delamination shapes. The splitting of the faces is referred to as the "imprint operation" by the CAD tool. Any layer that was created by the CAD Model Builder can be used for this imprinting operation. Then, the framework identifies the different face pieces where the cohesive or contact boundary conditions can be applied.

Separate boundary conditions to perform the structural analysis are created based on user input. These includes faces that need to be fixed and locations where a force boundary condition needs to be applied. These faces are identified using reference points and normals, which are directly passed to the FEA software. In addition, addition paran sis.

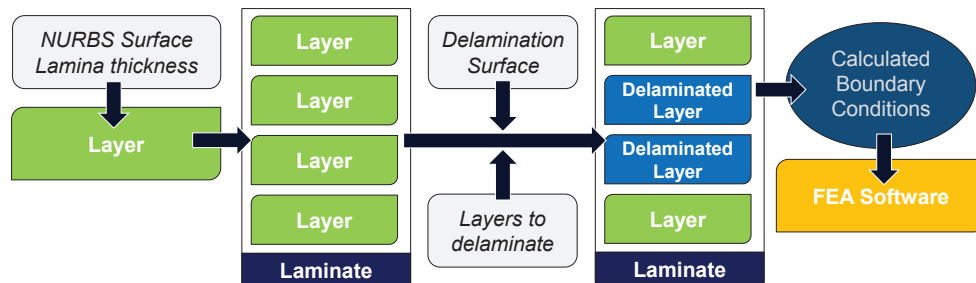


FIGURE 4. A flowchart explaining the steps of the delamination analysis currently defined in the automated framework.

RESULTS

The framework can be used to automatically create the composite laminates using a layer-by-layer structure, incorporate the defect data obtained from the UT scans, and analyze the final composite using SIMULIA Abaqus FEA software suite using a single Python script file.

Creating 3D Structural Models of the Composites

The CAD Model Builder can create 3D structural models of composites. It uses multiple layers that correspond to the laminae to build the final structural model. Depending on the model, the user can create multiple layers with different thickness to match the composite structure. The CAD Model Builder takes an input NURBS surface as the mold and a thickness value for extrusion of the mold surface. The mold surface can be a NURBS surface or a surface of a previously-created layer. Figure 5 shows two 8-layered 3D composite model created by the CAD Model Builder using planar and curved NURBS surfaces.

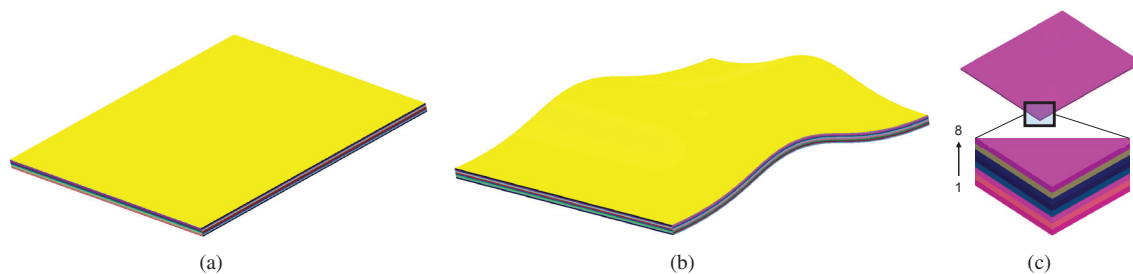


FIGURE 5. 8-layer composite models created by the CAD model builder: (a) a CAD model created using a planar NURBS surface, (b) a CAD model created using a curved NURBS surface, (c) Zoomed-in CAD model showing the different layers in different colors.

UT NDE Sample and Details of the Delaminated Layer

Our initial focus is on incorporating delamination defects into the structural models. We used a 32-layered CFRP sample that was impact damaged as our initial test sample. The sample was analyzed using bi-directional UT scan. The shape and location of the region of interest with respect to the laminate structure is used to correctly register the scan. Figure 6(a) shows the composite sample used and the 2x2" UT scan region.

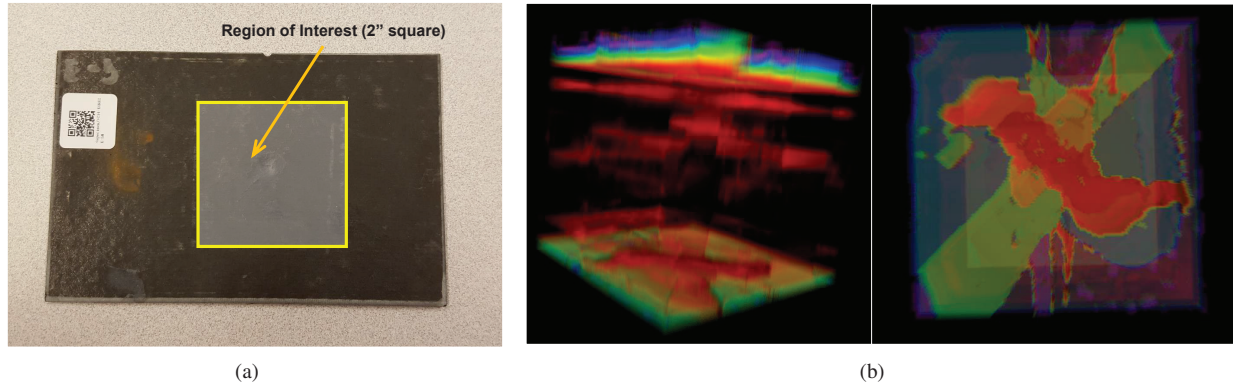


FIGURE 6. (a) A photograph of the composite sample used to obtain delamination data indicating the area where the UT scan was run, and (b) the volumetric rendering of the composite sample indicating the delaminated region in red on the top view.

In order to better visualize the delamination shape inside the composite lamina, we used a volumetric renderer. The data coming from the UT scan is front-wall corrected using the methods described under the Methods section. To obtain smooth waveform values, Hilbert transform is applied to the front-wall corrected data. Finally, the Hilbert-transformed data is rendered using ray-casting on the GPU. Figure 6(b) shows the volumetric renderer results showing the location of the delamination inside the laminate (left) and the shape of the top delamination used as an example in this paper (in red, right).

Adding Delaminations to the 3D Structural Model

The CAD Model Builder can create delaminated layers using the delamination data from UT scans or a simulation engine. The user can choose to add a single or multiple delaminated surfaces between any layers. Currently, the delamination shape is read from a CSV file that is output after processing the NDE data. This file includes the x, y, and z coordinates of the delamination shape. The CAD Model Builder creates the required geometrical shapes for cohesive, contact and no-model zones using the shape and imprints these shapes between the chosen layers.

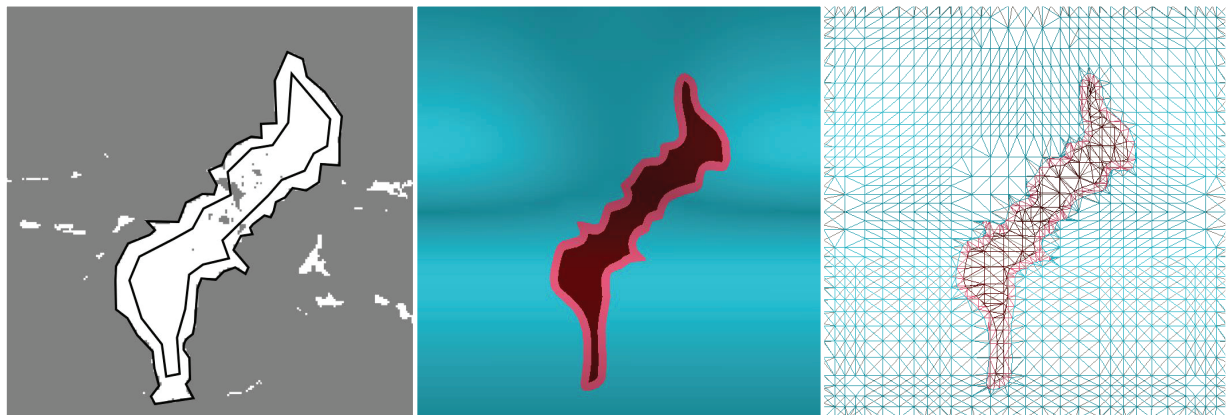


FIGURE 7. Visualization of the delamination model using the actual delamination image (left) and in CAD model builder (middle), meshed delamination surface (right).

Finite Element Analysis of the 3D Structural Model

The FE Model Builder can read the resulting CAD model from the CAD Model Builder and pass them along with the required boundary conditions to the FEA software. FE model builder applies the cohesive boundary conditions to construct the bonding between the layers and applies contact boundary conditions in the delaminated regions. In addition, the boundary conditions required to fix some of the faces and apply forces are also passed to the FEA software.

Figure 8 shows the visualization of the delamination model in SIMULIA Abaqus FEA software suite (left) and the deformed model after static bending (right). It is possible to observe the non-uniformity of the displacement field due to the delaminations present in the structure.

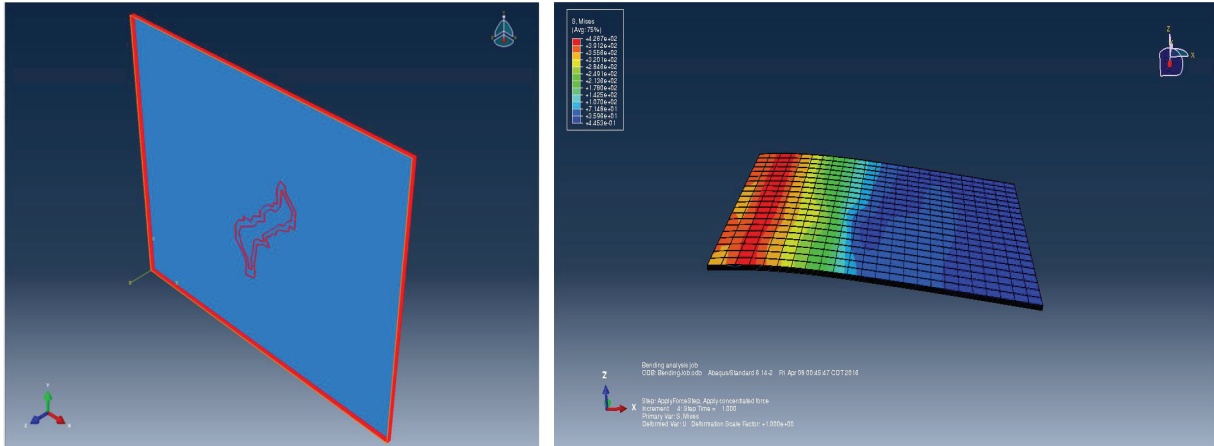


FIGURE 8. Outputs of the FE Model Builder in SIMULIA Abaqus FEA software suite showing the delamination shape (left) and the results of a bending analysis (right).

CONCLUSIONS

We have presented a proof-of-concept, automated framework for building composite laminates with delaminations for structural analysis. Using our framework, a designer can easily generate a complicated composite structure that can be directly used for structural analysis. The framework automates the model setup process, thereby removing tedious operations needed for setting up the boundary conditions in the model before analysis can be performed. In addition, we can incorporate a complicated delamination shapes obtained from the raw UT scan data of the composite between the corresponding layers of the laminate. Incorporating such high-fidelity damage models can dramatically improve the accuracy of residual strength predictions.

Future work on the framework will focus on incorporating complex composite elements, such as stiffeners, into the 3D structural models of the composites. In addition, we plan to incorporate methods to handle different kinds of defects, such as fiber breakage in the composite models.

ACKNOWLEDGEMENTS

This work is supported by National Aeronautics and Space Administration (NASA) under contract number NNL15AA12C through Center for Non-Destructive Evaluation (CNDE) at Iowa State University and, is a part of NASA Advanced Composites Project. We would like to thank Dr. William P. Winfree and Cheryl A. Rose from NASA for their valuable input and support for this project. We would also like to thank Dr. Ronald Roberts from CNDE at Iowa State University for the UT scans and the initial data.

REFERENCES

1. W. Deng, L. Shark, B. Matuszewski, J. Smith, and G. Cavaccini, [Insight-Non-Destructive Testing and Condition Monitoring](#) **46**, 157–161 (2004).
2. Z. Liu, D. S. Forsyth, J. P. Komorowski, K. Hanasaki, and T. Kirubarajan, [IEEE transactions on Instrumentation and Measurement](#) **56**, 2435–2451 (2007).
3. 3D ACIS Modeler, “<https://www.spatial.com/products/3d-acis-modeling/>,” (2016), accessed Oct 2016.
4. Abaqus FEA, “<http://www.3ds.com/products-services/simulia/products/abacus/>,” (2016), accessed Oct 2016.
5. W. Cantwell and J. Morton, [Composites](#) **22**, 347–362 (1991).
6. G. Alfano and M. Crisfield, [International journal for numerical methods in engineering](#) **50**, 1701–1736 (2001).
7. U. Ayachit, *The ParaView Guide: A Parallel Visualization Application* (Kitware, 2015).
8. Spatial Corporation, *3D ACIS Modeler Product Documentation* (2016), accessed Oct 2016.
9. Dassault Systemes, *Abaqus 6.14 Installation and Licensing Guide* (2014), accessed Oct 2016.
10. T. L. Cottom, [Computing in Science and Engineering](#) **5**, 88–97 (2003).
11. M. Garton, T. A. Gray, R. B. Thompson, and I. Gray, in *Review of Progress in Quantitative Nondestructive Evaluation*, (American Institute of Physics **1211**, Melville NY) **29**, 2141-2148 (2009).